DHI-WASY Software

FEFLOW® 6.1

Finite Element Subsurface Flow

& Transport Simulation System

Installation Guide & Demonstration Exercise

DHI-WASY GmbH

Waltersdorfer Straße 105, 12526 Berlin, Germany

Phone: +49-(0)30-67 99 98-0,

Fax: +49-(0)30-67 99 98-99

E-Mail: mail@dhi-wasy.de

Internet: www.feflow.com

www.dhigroup.com

Contents

I Installation Guide

I.1 Introduction

The FEFLOW simulation package contains the following main programs along with additional software tools:

FEFLOW® 6.1

FEFLOW is an interactive finite-element simulation system for modeling 3D and 2D flow, mass and heat transport processes in groundwater and the vadose zone.

FEFLOW is provided on DVD for the following platforms:

32-bit operating systems

- Windows XP, Vista, 7, Server 2003, Server 2008
- Linux: CentOS 5.1 (RedHat family), OpenSUSE 11.0 (SUSE family), Ubuntu 10.04 (Debian family)

64-bit operating systems

- Windows XP x64 Edition, Vista x64 Edition, 7 x64 Edition, Server 2003 x64 Edition, Server 2008 x64 Edition
- Linux: CentOS 5.1 (RedHat family), OpenSUSE 11.0 (SUSE family), Ubuntu 10.04 (Debian family)

FEFLOW for other Linux distributions may be available for download from the FEFLOW website *www.feflow.com*. If you need FEFLOW for another Linux distribution, please do not hesitate to contact us at *support@dhi-wasy.de*!

For evaluation purposes, it is possible to obtain a fully functional but time-limited license from DHI-WASY, one of the DHI offices, or from your local FEFLOW distributor.

FEFLOW® Viewer

FEFLOW Viewer is free software for visualizing FEFLOW models and results and for postprocessing purposes. FEFLOW Viewer is installed with FEFLOW.

WGEO® 6.0

WGEO® is a sophisticated georeferencing, geoimaging and coordinate transformation software developed by DHI-WASY GmbH. A license for WGEO® Basis and the flexible 7-parameter transformation comes with each FEFLOW license and is installed automatically.

WGEO® is provided for the Windows platform.

I.2 Installing FEFLOW (Windows)

I.2.1 Introduction

FEFLOW 6.1 provides powerful state-of-the-art editing, visualization and evaluation tools as well as computational methods for most kinds of groundwater models.

To run FEFLOW in single-seat mode, the DHI-WASY License Manager NetLM has to be installed locally.

If the the license shall be acquired from a license server using a network connection (network license required), no local installation of NetLM is necessary.

A license server can be set up by installing NetLM on that machine. The installation of FEFLOW on the same computer is not necessary.

A typical FEFLOW installation on a Windows operating system consists of three steps:

- Installation of FEFLOW and additional programs
- Installation of the demo data package
- Installation of the DHI-WASY license manager NetLM

After inserting the DVD into the DVD drive, an overview of the DVD contents is shown automatically. If autostart is disabled, run **Starter.exe** from the windows directory on the DVD.

The hyperlinks in the overview can be used to start the different parts of the FEFLOW installation, to view the documentation and example movies, and to install third-party software.

FEFLOW is automatically installed as a 32 bit version on 32 bit systems, and as 32 and 64 bit versions on 64 bit operating systems.

I.2.2 System recommendations

The following system specifications are recommended as a minimum configuration. The memory requirements depend on the size and complexity of the actual model to be simulated.

- 512 MB RAM
- 250 MB of disk space
- Dedicated graphics card with up-to-date graphics driver

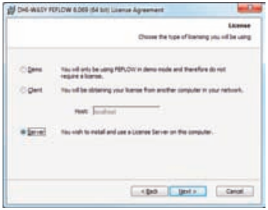
I.2.3 FEFLOW Installation

Start the ***Windows Installer*** by clicking on the hyperlink ***FEFLOW Program Files***. Click ***Next*** after each step to proceed to the next step.

1. A ***Welcome*** screen appears first.

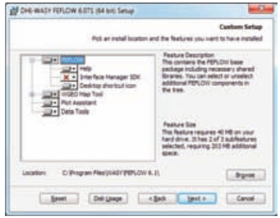
2. In the next step, the ***License Agreement*** has to be accepted. Please read it carefully before proceeding with the installation.
3. Wait until the installer has determined the available disk space.

4. In the ***License*** window, choose ***Demo*** for testing FEFLOW without a license. Select ***Client*** if the license to be used is installed on a remote license server (Network License) and type in the name or IP number of the license server. Choose ***Server*** if a Single Seat License is to be used or if the machine is intended to act as a license server for a Network License.

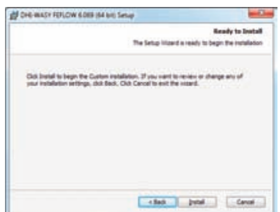


5. Select the packages to install. Details about the packages can be found on the right of the win-

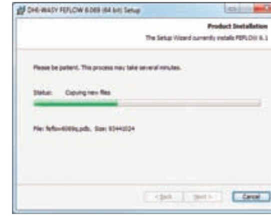
dow and on page **10** of this booklet. The default installation location is C:\Program Files\WASY\FEFLOW 6.1. For specifying a different destination directory, click ***Browse***.



6. Start the installation or go back to change the settings.



7. FEFLOW is installed. This may take several minutes.



8. Finish the installation by clicking ***Finish***.

I.2.4 Demo Data Installation

The demo data installation is started by clicking on the hyperlink ***FEFLOW Demo Data***.

I.2.6 Installation of the Network License Manager NetLM

Before installing the Network License Manager NetLM we recommend to deinstall all previous versions of NetLM.

The installation is started by clicking on the hyperlink ***License Manager NetLM***. Follow the instructions in the installation dialog.

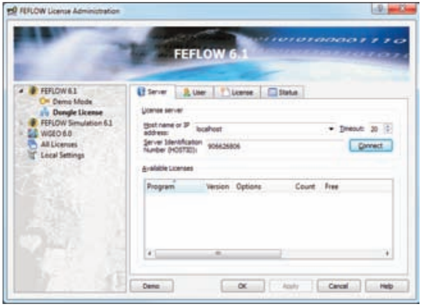
I.2.7 License installation

This step can be skipped for working in Demo mode only. All license installations have to be done with administrator privileges.

There is only one dongle for each copy of FEFLOW. If the dongle is lost, it can only be replaced by purchasing a new license of FEFLOW!

Make sure that the firewall allows (local) TCP/IP connections on port 1800.

- Attach the dongle (hardware lock) to the USB port. Start the ***DHI-WASY License Administration*** tool by clicking on the ***WASY License Administration*** entry in the All Programs\WASY group of the Windows start menu.



- In the tree view on the left side, select ***Dongle license*** in the FEFLOW 6.1 branch. In the ***Hostname or IP address*** field, insert ***localhost*** if the dongle is installed locally, or insert the name or the IP number of the remote license manager.
- Click ***Connect***. Check whether the number returned in the field ***HOSTID*** is identical to the number on the FEFLOW license sheet. In case that multiple dongles of the same brand are connected, a HOSTID mismatch may occur. In this case remove all dongles except the DHI-WASY dongle.
- Switch to the ***Licenses*** tab and enter the license information from the FEFLOW license sheet.

For the 64 bit version of FEFLOW, a HASP HL USB dongle is required. If you do not already have such a dongle, please contact your FEFLOW distributor or DHI-WASY for a dongle exchange!

The license information can be pasted from the clipboard if the license has been received digitally. Copy the selected section in the license document to the clipboard and use the ***Paste license from clipboard*** button to insert all the license information at once. Please also make sure that the same license type as on the license sheet is selected:

- Single seat license – FEFLOW can be run only on the computer the dongle is attached to.
- Network license – FEFLOW can be used on any computer connected to the license server via TCP/IP network (LAN or WAN/internet).

For the version, please note that you have to use "6.1x" because your license is valid for all sub-versions of FEFLOW 6.1.

- Click ***Install***. A message box indicates that the license has been successfully installed. If the installation was not successful, check all the license information in comparison to the information provided on the license sheet. The information has to be identical in all details.
- Click ***OK*** to close the ***License Administration*** dialog.
- Choose FEFLOW 6.1 from the WASY program group or double click on the desktop icon to start FEFLOW.

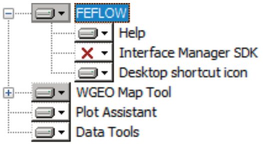
Using a Network License for FEFLOW, the WASY License Manager can be installed on any computer within the network (LAN or WAN) without the complete FEFLOW installation. Clients need TCP/IP connection on port 1800 to have access to the license server.

I.3 Installing FEFLOW (Linux)

- Browse to the linux directory on the DVD.
- Browse to the sub directory corresponding to your Linux distribution.
- For a full installation, use the following command: `rpm -i *.rpm`
- For deinstalling all WASY packages, use `rpm -e `rpm -qa | grep '^wasy-'``
- Note that you may need root privileges to perform these commands.

I.4 Installation Packages

- Packages for installation can be selected during the first installation or by re-running the installation in ***Modify*** mode.
- A description for each package is shown in the right part of the ***Select feature*** dialog of the installation by selecting one of the packages.
- The following packages are available:



FEFLOW

FEFLOW program files - required for running FEFLOW

- Help - FEFLOW help system
- Interface Manager SDK - development kit for the open programming interface IFM (required for plug-in development)
- Desktop shortcut icons on the Windows desktop

WGEO

WGEO Basis is licensed automatically with FEFLOW. If a license dialog show up, just click on Cancel.

Georeferencing, geoimaging and transformation software - a WGEO license is installed automatically.

- WGEO help - online help system
- German Transformations - coordinate transformation routines for Germany (may require separate licensing)
- Desktop shortcut icon - WGEO icon on the Windows desktop

Plot Assistant

GIS-like software for producing plots with FEFLOW data.

Data Tools

Scripts for data checking and format conversion.

In the ***demo data*** installation there are the following packages:

- Examples - example models
- Exercise - data for the demonstration exercise
- Tutorial - data for the tutorials (User Manual)
- Benchmarks - benchmark models

II Demonstration Exercise

II.1 Introduction

II.1.1 About FEFLOW

FEFLOW (Finite Element subsurface FLOW and transport system) is an interactive groundwater modeling system for

- three-dimensional and two-dimensional
- areal and cross-sectional (horizontal, vertical or axisymmetric)
- fluid density-coupled, also thermohaline, or uncoupled
- variably saturated
- transient or steady state
- flow, mass and heat transport
- reactive multi-species transport

in subsurface water resources with or without one or multiple free surfaces.

FEFLOW can be efficiently used to describe the spatial and temporal distribution and reactions of groundwater contaminants, to model geothermal processes, to estimate the duration and travel times of chemical species in aquifers, to plan and design remediation strategies and interception techniques, and to assist in designing alternatives and effective monitoring schemes.

Sophisticated interfaces to GIS and CAD data as well as simple text formats are provided.

The option to use and develop user-specific plug-ins via the programming interface (Interface Manager IFM) allows the addition of external code or even external programs to FEFLOW.

FEFLOW is available for WINDOWS systems as well as for different Linux distributions.

Since its first appearance in 1979 FEFLOW has been continuously extended and improved. It is consistently maintained and further developed by a team of experts at DHI-WASY. FEFLOW is used worldwide as a high-end groundwater modeling tool at universities, research institutes, government agencies and engineering companies.

For additional information about FEFLOW please do not hesitate to contact your local DHI office, one of the FEFLOW distributors, DHI-WASY, or have a look at the FEFLOW web site *http://www.feflow.com*.

II.1.2 Scope and Structure

This exercise provides a step-by-step description of the setup, simulation, and post processing of a three-dimensional flow and transport model based on (simplified) real-world data, showing the philosopy and handling of the FEFLOW user interface.

Demonstration Exercise

You can skip any of the steps in this exercise by loading already prepared files at certain stages. These files are not necessarily ready to run in the simulator.

The demonstration exercise is not intended as an introduction to groundwater modeling itself. Therefore, some background knowledge of groundwater modeling is required, or common literature should be consulted in parallel.

The exercise covers the following work steps:

- Definition of the basic model geometry
- Generation of a 3D finite-element mesh
- Setup of a transient transport model, including initial conditions, boundary conditions and material properties
- Import of GIS data and regionalization
- Simulation run
- Results visualization and post processing

II.1.3 Terms and Notations

In addition to the verbal description of the required screen actions this exercise makes use of some icons. They are intended to assist in relating the written description to the graphical information provided by FEFLOW. The icons refer to the kind of setting to be done:

For following the exercise, the demo data files for FEFLOW have to be installed. The demo data installation package is available on the FEFLOW DVD as well as on the FEFLOW web site for download.

- main menu
- context menu
- toolbar
- panel
- button
- input box for text or numbers
- switch toggle
- radio button
- checkbox

All file names are printed in **bold red**, map names are printed in *red italic* and numbers or text to be entered by the user in **bold green**. Keyboard keys are referenced in <*italic*> style. All required files are available in the FEFLOW demo data. The symbol indicates an intermediary stage where either a prepared file can be loaded to resume this exercise or - if working with a license - the model can be saved. Thus the exercise does not have to be done in one step even in demo mode.

II.1.4 Requirements

If not already done, please install the FEFLOW software including the demo data package. A license is not necessary to run this tutorial (FEFLOW can be run in demo mode).
The latest version of FEFLOW can be downloaded from the website *www.feflow.com*. In case of any problems or additional questions please do not hesitate to contact the FEFLOW technical support (*support@dhi-wasy.de*).

II.1.5 Model Scenario

A fictitious nitrate contaminant has been detected near the small town of Friedrichshagen, in the southeast of Berlin, Germany. An increasing concentration can be observed in two water supply wells. There are two potential sources of the contamination: The first are abandoned sewage fields close to a waste-water treatment plant located in an industrial area northeast of town. The other possible source is an abandoned waste-disposal site further east.

A three-dimensional groundwater flow and contaminant transport model is set up to evaluate the overall threat to groundwater quality, and to quantify the potential pollution. First, the model domain needs to be defined. The town is surrounded by many natural flow boundaries, such as rivers and lakes. There are two small rivers that run north-south on either side of Friedrichshagen that can act as the eastern and western boundaries. The lake Müggelsee can limit the model domain to the south. The northern boundary is chosen along an northwest-southeast hydraulic contour line of groundwater level north of the two potential sources of the contamination.

The geology of the study area is comprised of Quaternary sediments. The hydrogeologic system contains two main aquifers separated by an aquitard. The top hydrostratigraphic unit is considered to be a sandy unconfined aquifer up to 7 meters thick. The second aquifer located below the clayey aquitard has an average thickness of approximately 30 meters.

The northern part of the model area is primarily used for agriculture, whereas the southern portion is dominated by forest. In both parts, significant urbanized areas exist.

II.2 Getting Started

II.2.1 Starting FEFLOW

On Windows Systems

- Start FEFLOW 6.1 via the corresponding desktop icon or the startup menu entry.

On Linux Systems

- Type feflow61q in a console window and press *<Enter>* .

If no FEFLOW license is available, FEFLOW asks whether to start in demo mode. The demo mode does not allow loading and saving of files with more than 500 nodes. Specially prepared demo files coming with FEFLOW are an exception. Such files are provided for this example so that the model setup can be interrupted and picked up again.

II.2.2 FEFLOW 6.1 User Interface

The user interface components are organized in a main menu, toolbars, panels, view windows, and dialogs.

While the main menu is always visible, the other parts of the interface can be customized, adding or hiding particular toolbars and panels by using the menu command ***View*** > ***Toolbars*** and ***View*** > ***Panels***, respectively. Please keep in mind that not all panels and toolbars are displayed by default. Thus this exersice may require to access a function in a toolbar or panel that is not visible at that moment. The toolbar or panel has to be added then.

During the work with FEFLOW models, view windows display a certain type of view on the model and its properties. There are different types of view windows: ***Supermesh*** view, ***FE-Slice*** view, ***3D*** view, ***Cross-Section*** view and ***Data-Trace*** view. The availability of different functionality like toolbars depend on the currently active view type.

View windows can be closed via the corresponding button in the view frame. New view windows can be opened by selecting ***Window*** > ***New*** and choosing the respective view window type.

The last type of user interface component relevant for the exercise are charts. Looking very similar to panels, they contain plots of time curves. Missing chart windows can be added to the user interface

In this exercise, different file types are used as data source at the different stages of modelling to show the number of options. In practical projects, it may be preferred to store basic data in one file type, e.g., shp when using GIS.

by opening ***View*** > ***Charts*** from the menu and choosing the required chart type from the list.

Last, but not least it might be worth to mention that all steps done in FEFLOW can be undone and redone via the corresponding toolbar buttons. There is no limit on the number of undo steps.

II.3 Geometry

II.3.1 Maps and Model Bounds

After opening FEFLOW, start a new model by using the menu command ***File*** > ***New*** or the ***New*** button in the ***Standard*** toolbar.

The first step of model setup is the definition of the ***Initial Domain Bounds***. This can be done manually, or by loading georeferenced maps.

All necessary files for this exercise are provided with the FEFLOW Demo Data package and are located in the project folder ***demo/exercise***. The map files are found in the subdirectory ***import+export***.

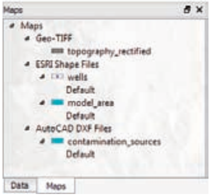
Click on ***Multiple, possibly nonoverlapping maps [...]*** and press ***Finish***. Load all the following maps at once (by holding *<Ctrl>* on the keyboard) to ensure that FEFLOW uses the bounding box of all the maps to define the initial domain bounds. You may have to select ***All Maps*** in the ***Files of Type*** selector at the bottom of the dialog.

The particular map files that are needed now are:

- **topography_rectified.tif** (a georeferenced raster image of the model area for better orientation)
- **model_area.shp** (a polygon map that contains several polygons denoting the outer model boundary and embedded contamination areas)
- **contamination_sources.dxf** (the footprint of the sewage fields and the waste disposal as polygons)
- **wells.shp** (the positions of the wells)

Some of these maps will also be used for model parameterization later on.

After import, the maps are listed in the ***Maps*** panel, sorted by their file type (see figure). A double click on the Geo-TIFF *topography_rectified* adds the georeferenced topographic map to the active ***Supermesh*** view window.

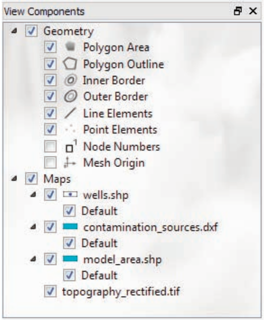


Except for the maps *contamination_sources* and *topography_rectified* all other maps are ESRI shape files. These vector files occupy their own branch in the

tree, each with a default map layer. Double-click on all the ***Default*** layer entries to add the visualization layers of the shape files to the ***Supermesh*** view.

Now have a closer look at a second panel, the ***View Components*** panel. This panel lists the components that are currently plotted in the active view window.

When loading the map layers to the view, the maps have also been added to the tree in the ***View Components*** panel.



The drawing order of maps can be modified by dragging them with the mouse to another position in the tree (this might become necessary as the model area polygon may overlay the polygons of the mass sources). The topmost map is drawn on top.

To switch a map on and off, the checkbox in front of the map name can be checked/unchecked. If checking/unchecking the checkbox of an entire branch all the maps in this branch become visible/invisible at the same time.

The topographic map has mainly been loaded for providing a regional context. For more clarity, it can be switched off before starting with the following operations. Make sure that the other maps are visible.

II.3.2 Supermesh

In the simplest case, the supermesh contains a definition of the outer model boundary. In addition, geometrical features such as the position of pumping wells, the limits of areas with different properties or the courses of rivers can be included to be considered for the generation of the finite-element mesh. Additionally, the polygons, lines and points specified in the supermesh can be used later on to assign boundary conditions or material properties.

As mentioned above, a supermesh may contain three types of features:

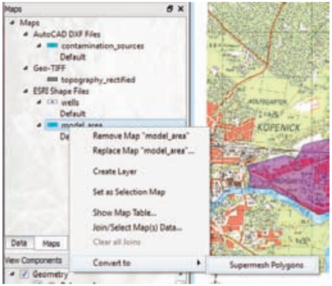
- polygons
- lines
- points

At least one polygon has to be created to define the model area boundaries.

The editing tools are found in the *Mesh Editor* toolbar:

Outer boundary and contamination sources

The polygons can be directly loaded from the map *model_area*. In the *Maps* panel, open the context menu of this map (with a right-click on the map name) and choose *Convert to* > *Supermesh Polygons*.

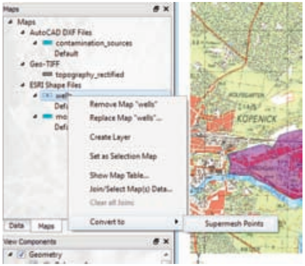


The supermesh imported from the map does consist of several adjacent, non-overlapping polygons. For additional editing, different tools are available to manually modify the supermesh.

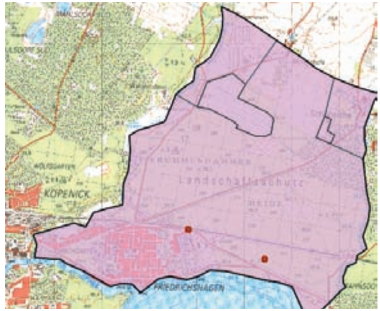
Before generating the finite-element mesh, the well locations are to be included in the supermesh for this example.

Well locations

The positions of the wells can be imported directly from a map as well. Open the context menu of the map *wells* in the *Maps* panel and choose *Convert to* > *Supermesh Points*.



The points immediately appear as red dots in the *Supermesh* view.



exercise_fri1.smh

II.3.3 Finite element-mesh

Once the outer boundary and other geometrical constraints have been defined in the supermesh, the finite-element mesh can be generated.

All necessary tools can be found in the ***Mesh Generator*** toolbar.

First, one of the mesh generation algorithm provided by FEFLOW is chosen from the drop-down list in the ***Mesh Generator*** toolbar.

For this example, choose ***Gridbuilder***. Click ***Generate Mesh*** to start mesh generation.

A new ***Slice*** view is automatically opened, depicting the resulting finite-element mesh.

For our purpose, especially for the simulation of contaminant transport, this initially generated mesh does not seem to be appropriate. A finer spatial resolution is required.

Activate the ***Supermesh*** view again so that the ***Mesh Generator*** and ***Supermesh*** toolbar become visible again.

In the ***Mesh Generator*** toolbar, enter **3000** as ***Total Elements*** and click ***Generate Mesh*** again. The finite-element mesh in the ***Slice*** view is updated, showing a finer discretization now.

Local refinement

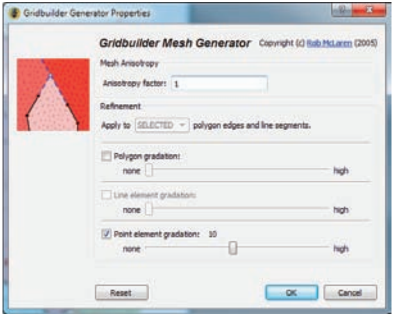
At the pumping wells steep hydraulic gradients are expected at the center of the well cone. To realistically represent these, fine discretization is necessary, too.

Switch to the ***Supermesh*** view again to see the ***Mesh Generation*** toolbar. If the view has been accidentally closed, re-open it by choosing ***Window*** > ***New*** > ***Supermesh View*** from the menu.

Zooming functions can be used at any time. Press and hold the right mouse button, move the mouse up/down). Pan by pressing and holding the mouse wheel and moving the mouse to any direction. Also the mouse wheel may be used for zooming.

Besides refinement at points or polygon borders, FEFLOW also provides the means to edit the desired relative mesh density on a polygon-by-polygon basis.

Click on ***Generator Properties*** to open the ***Generator Properties*** dialog.

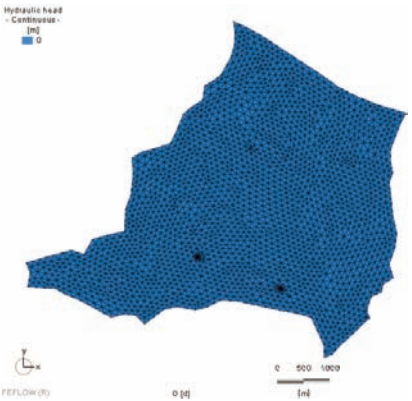


To obtain a refinement around the well locations, activate the ***Point element gradation***. Apply a refinement level of **10**.

Leave the dialog by clicking on ***OK***.

Click ***Generate Mesh*** one last time and check the changes in the mesh. The refinement pattern looks similar to the one in the figure where the mesh around the well locations is finer.

exercise_fri1.fem



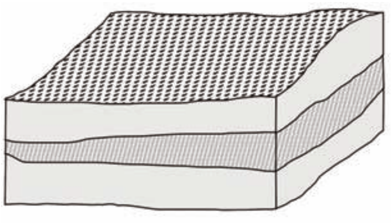
II.3.4 Expansion to 3D

Up to this point we have worked on the model seen in top view, not considering the vertical direction. Starting from this 2D geometry, a 3D model consisting of several layers is set up.

The actual elevation of the layers tops and bottoms is derived by an interpolation based on map data (point-based data).

For this example, three geological layers are considered for the model. An upper aquifer is limited by the ground surface on top and by an aquitard at the bottom. A second aquifer is situated below

the aquitard, underlain by a low permeable unit of unknown thickness. This underlying stratigraphic layer is assumed to be impervious and is not part of the simulation.

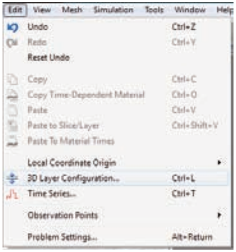


FEFLOW distinguishes between layers and slices in 3D. Layers are three-dimensional bodies that typically represent geological formations like aquifers and aquitards. The interfaces between layers, as well as the top and bottom model boundary are called slices.

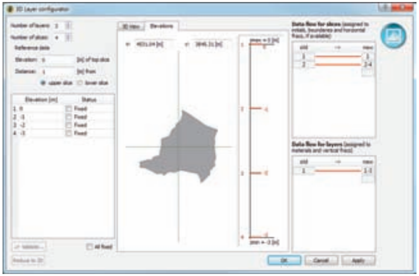
In a first step, the numbers of layers and slices are defined. The actual stratigraphic data are applied in a separate step afterwards.

Initial 3D Setup

Open ***Edit*** > ***3D Layer Configuration***.



In the upper left corner of the dialog, a text field shows the current number of layers (1). Increase this value to **3** and hit <*Enter*>. This makes FEFLOW switch the model geometry to 3D, the model containing 3 layers. The number of slices automatically changes to 3 + 1 = 4 . By default, the top slice has a spatially constant elevation of 0 m. The other slices are placed below with a distance of 1 m each.



Click on OK to apply the settings and to exit the dialog. After finishing the basic layer configuration of the 3D model, a ***3D*** view automatically opens. This view shows the actual 3D geometry of the model, now containing 4 planar slices with a distance of 1 m each. The ***3D*** view background by default is set to black. To achieve better visibility in print, for all images in this exercise a white view background has been applied.

exercise_fri2.fem

Elevation Data

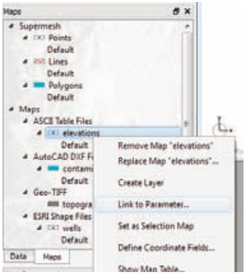
This raw geometry will be formed into its real shape by regionalizing elevation data contained in map files.

The basic data have been derived from a DEM and from borehole logs, and have been combined into a text file (extension dat, tab separated) with four columns: X, Y, Ele, and Slice. Such a file can be edited in a text editor, or in spreadsheet software such as Microsoft Excel or Open Office / LibreOffice. Containing the target slice number as a point attribute, the file can be used as the basis for regionalization of elevations for all slices at once. The elevations are given in meters ASL.

X	Y	Ele	Slice
3403999.5	5818015	39.5	1
3403999.5	5818405.5	39.5	1
3403999.5	5819254	39.5	1
3403999.75	5814206	34.5	1
3403999.75	5813779.5	37	1
3403999.75	5813957.5	37	1
3403999.75	5814522.5	37	1
3403999.75	5815095	37	1

The file has to be loaded as a map before its attribute data can be used as the basis for interpolation. Go to the ***Maps*** panel and use the context menu (***Add Map(s)...***) to add **elevations.dat** to the list of loaded maps. It is not necessary to visualize the map in the view.

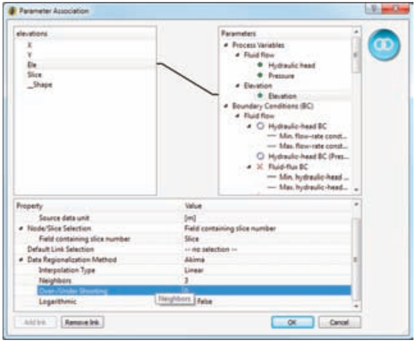
As a next step, the attribute values of the data file need to be associated with (linked to) their respective FEFLOW parameter, in this case with the elevation. In order to do this, open the context menu of the map *elevations* with a right click and choose ***Link to Parameter...***.



On the left-hand side of the dialog, the available attributes of the map are listed. Select the entry ***Ele*** with a mouse click.

On the right-hand side, a tree view contains all available FEFLOW parameters that can be associated with the data. In this tree, open the ***Process Variables*** > ***Elevation*** branch and click on ***Elevation***.

Click on Bu ***Add Link*** to establish a connection between the values in the map and the elevation data, or - alternatively - double click on ***Elevation*** to set the link.



Besides linking the attribute field to the model properties, a number of settings has to be done to ensure an appropriate regionalization when importing the map data to the nodal values.

By default, FEFLOW expects elevation data to be in the unit meters, which is correct in this case.

FEFLOW only applies two-dimensional interpolation. To separate data for the different slices, select ***Field containing slice number*** in ***Node/Slice Selection***. In the next line, choose the attribute ***Slice*** as ***Field containing slice number*** (see image).

To transfer data from the points in the map to all the mesh nodes, a regionalization method has to be applied. From the dropdown menu for ***Data Regionalization Method*** in the lower part of the dialog, choose the ***Akima*** method. As the properties, set:

- ***Interpolation type***: ***Linear***
- ***Neighbors***: **3**. Only the three map points that are closest to a mesh node are used for the interpolation.
- ***Over-/Under Shooting***: **0**. Thus the resulting values may not exceed the range of input values.

Click on ***OK*** to apply these settings and to close the dialog.

Elevation data assignment

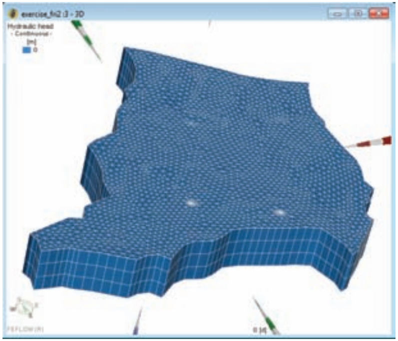
Click into the ***3D*** view to bring it to front. Make sure to have the ***Rotate*** tool in the ***View*** toolbar activated. Rotation, panning, and zooming can be easily performed using the mouse:

- Left mouse button: rotate the model around its center of gravity
- Center mouse button (mouse wheel): pan the model (left/right/up/down)
- Right mouse button: zoom (in/out)

As the model has a rather small vertical extent compared to its horizontal dimensions, the (vertical) z-axis should be exaggerated.

This can be done in the ***Navigation*** panel (***View*** > ***Panels***).

Click on the tab ***Projection*** and move the ***Scaling*** slider bar upwards until you have achieved a convenient view on the 3D model. Alternatively, the <*Shift*> key in combination with the mouse wheel also changes the stretch factor for the active ***3D*** view.

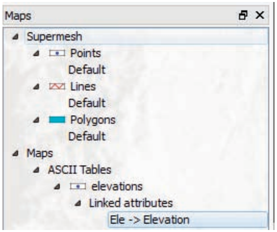


Demonstration Exercise

As a next step, the target locations of the data assignment have to be defined, i.e., all nodes in the mesh have to be selected.

To finally assign the elevation data by regionalization from map data to the selected nodes, two more steps are required:

- In the ***Maps*** panel, open the branch ***Maps*** > ***ASCII Table Files*** > *elevation*. Under ***Linked Attributes***, double-click on ***Ele*** -> ***Elevation***.



- Hereby, in the ***Editor*** toolbar, the map *elevation* is automatically set as data source in the input box and the model property ***Elevation*** is activated as parameter. Note that also ***Elevation*** has been chosen in the ***Data*** panel and is now shown in bold letters.

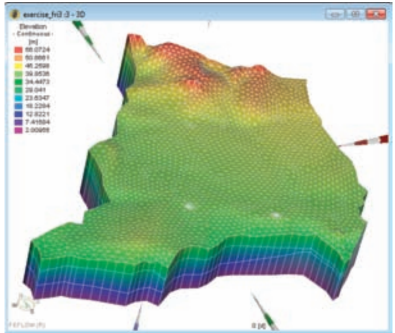
- Click on ***Select All*** in the ***Selection*** toolbar.

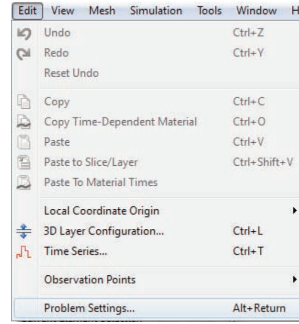
- Click ***Assign*** in the ***Editor*** toolbar to apply the new elevation data to all nodes.

In the ***3D*** view the node elevations are immediately updated.

Click on ***Clear Selection*** in the ***Selection*** toolbar.

The result looks as shown in the figure below. Probably the ***Scaling*** has to be adjusted again (***Navigation*** panel > ***Projection*** tab or <*Shift*> - mouse wheel) to account for the changed vertical extent.





exercise_fri3.fem

II.4 Problem settings

FEFLOW provides the means to simulate a number of different physical processes in different spatial and temporal dimensions, ranging from simple 2D steady-state flow models to transient, unsaturated, density-coupled reactive transport models. As the input parameters depend on the model type, the general problem settings are typically done in the beginning.

Go to ***Edit*** > ***Problem Settings*** to open the ***Problem Settings*** dialog, where all general settings related to the current model are done.

All these settings are organized in thematic pages controlled by a tree view on the left-hand side of the dialog.

Problem class

The principal type of the FEFLOW model is defined on the ***Problem Class*** page.

Below the ***Scenario description*** (which is not mandatory to be modified) one of two general types of problems – saturated media and unsaturated/variably saturated media is chosen.

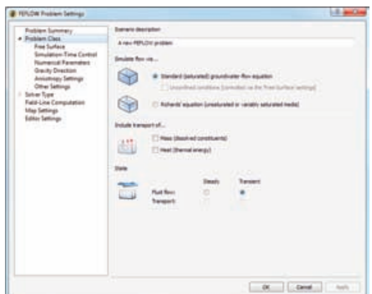
By default ***Standard (saturated) groundwater flow equation*** is selected, applying Darcy's equation. Though this option is selected, the model is able to account for phreatic conditions.

Additional selection options are described in detail in the FEFLOW help system.

The second option - ***Richards' equation (unsaturated or variably saturated media)*** - would lead to Richards' equation being applied, being capable of accounting for both saturated and unsaturated conditions within one model.

In this particular case, it is not expected that considering the unsaturated/variably saturated zone would change the model result to an extent that would justify the additional effort for the solution.

Keep the default setting (***Standard groundwater-flow equation***) for this exercise.



On the same page, ***Problem class*** settings are done. Besides choosing between transient and steady-state conditions, it is also possible to add solute or heat transport processes to the always executed flow simulation.

However, activating the transport option now would increase the complexity to an extent that is not necessary at this stage. Thus we first focus on the flow model and add mass transport at a later stage.

The flow model is set up for steady-state conditions, switch to ***Steady***.

Click on ***Apply*** to apply the changes.

Free Surface

The first aquifer in the simulation area is known to be unconfined, so a phreatic water table is to be simulated. For this example, an approximation of the phreatic level by applying a pseudo-unsaturated technology is chosen. Hereby, a reduced conductivity is applied during the simulation to model layers if they are located above the water table. For more information on the handling of free surfaces in 3D models, please refer to the help system and FEFLOW Documentation.

The settings for unconfined conditions are located on the ***Free Surface*** page. First of all, switch to ***Unconfined aquifers(s)***. In the ***Status*** column, open the drop down list of Slice 1 and choose the option ***Phreatic***.

For the slices 2 and 3 keep the option *Dependent* (the status of the bottom slice 4 is fixed and cannot be changed).

Finally, set the ***Residual water depth for unconfined layers*** to a value of **0.05** m to account for the residual water content in the unsaturated zone. In this way, finite elements above the water table have a small residual conductivity. This allows modelling a rerise of the water table and also increases the numerical stability of the model.

Close the dialog by clicking *Apply* and *OK*.

exercise_fri4.fem

II.5 Model Parameters

In the following sections, the physical properties of the study area are applied to the finite-element model.

The respective parameters are found in the *Data* panel. The parameters are organized in the following main branches in the tree view:

- *Process Variables*
- *Boundary Conditions*
- *Material Properties*
- *Auxiliary Data*
- *User Data*
- *Discrete Elements*

II.5.2 Boundary conditions

To calculate the hydraulic head distribution between the upstream and downstream boundary, appropriate boundary conditions are applied. For the sake of simplicity, they will be kept in a rather simple way:

- Southern border: The lake Müggelsee completely controls the head along the southern boundary. The lake water level of 32.1 m is used as the value for a 1st kind (Dirichlet) hydraulic-head boundary condition.
- Northern border: As there is no natural boundary condition like a water divide close to the boundary, a head contour line will be used instead (hydraulich head = 46 m).
- Western and eastern border: Two small rivers (the Fredersdorfer Mühlenfließ and the Neuenhagener Mühlenfließ) form the boundaries at the western and eastern limits of the model. As they follow roughly the groundwater flow direction, we assume these heavily clogged creeks to represent boundary streamlines. No exchange of water is expected over this boundary and therefore a no-flow boundary condition is assumed.
- Finally, two wells, with a pumping rate of 900 m^3/d and 1,000 m^3/d, respectively, are located in the southern part of the model. These represent a number of large well fields in reality.

The hydraulic head boundary conditions are entered manually, while the wells are derived from a map.

Manual editing is often easier if being done in a 2D view. Thus switch to the ***Slice*** view. If you have accidentally closed it, a new view can be opened via ***Window*** > ***Slice*** view.

Slice view

This view type always shows a single slice or layer. Browsing between the slices is easiest by hitting the *<Pg Up>* and *<Pg Down>* keys, respectively. Alternatively, the layer/slice to be seen in the view can be directly selected in the ***Spatial Units*** panel.

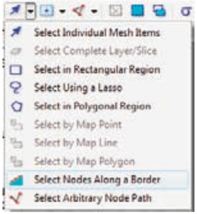
The recommended tool for navigation in the ***Slice*** view is the ***Pan*** tool in the ***View*** toolbar. The mouse buttons are associated with the following functions:

- Left and center mouse button: pan
- Right mouse button: zoom (in/out)
- Mouse wheel: zoom (in/out) in steps

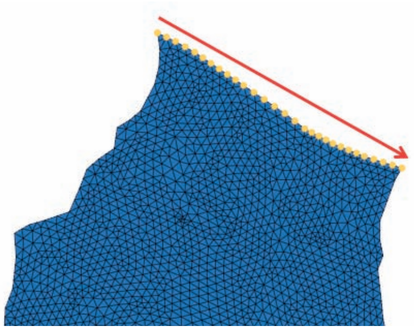
Northern Boundary

Zoom to the northern boundary.

For the ***Slice*** view, the ***Selection*** toolbar provides additional tools for selecting nodes compared to the ***3D*** view. Choose ***Select Nodes Along a Border***.

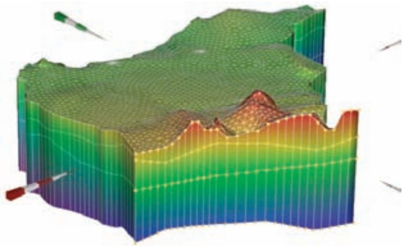


Having this tool selected, click on the westernmost node of the boundary and hold down the left mouse button, move the mouse cursor to the easternmost node and release the button. The nodes of the northern border are highlighted as yellow points. The selection is shown in the ***3D*** view simultaneously.

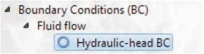


Next, the selection is extended to the other three slices of the model. A time-saving way to do this is the application of the ***Copy Selection to Layers/Slices*** tool. Start the tool and select all slices in the upcoming dialog (manually or by hitting *<Strg>-<A>)*. Click on ***OK***.

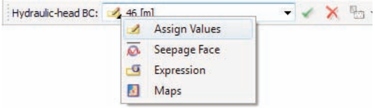
Bring up the ***3D*** view and ensure that indeed all nodes at the northern boundary are selected.



Go to the ***Data*** panel and double-click on ***Boundary Conditions (BC)*** > ***Fluid flow*** > ***Hydraulic-head BC***, the type of boundary condition to be applied.



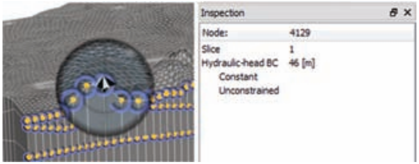
To manually assign a hydraulic-head boundary condition make sure that the ***Assign Values*** method is active in the ***Editor*** toolbar (see figure). If not, open the context menu by a right click on the symbol in the input box and choose ***Assign Values***.



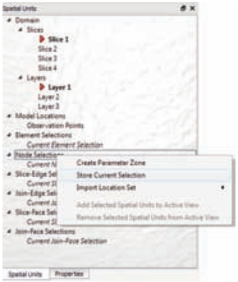
Enter 46 m in the input box of the *Editor* toolbar and click on the *Assign* button.

Blue circles appear around the selected nodes to indicate the ***Hydraulic-head BC***. While having the ***3D*** view active, double-click on ***Hydraulic-head BC*** in the *Data* panel to also show the boundary conditions in 3D.

The values of the boundary condition can be checked using the inspector tool, which is activated by clicking on ***Inspect nodal/elemental values*** in the *Inspector* toolbar. Move the hair-cross to a node with a boundary condition. The values of all properties currently visible in the active view are shown in the *Inspection* panel.



The inspector tool can be closed by hitting <*Esc*> or by activating another tool (e.g., *Rotate*) in the *View* toolbar.

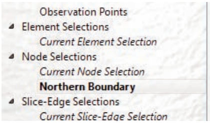


After defining the boundary condition at the northern boundary, we store the current selection for later use:

Open the context menu of the *Spatial Units* panel and choose ***Store Current Selection*** (will appear in the *Spatial Units* panel as ***Node Selection #1***). To give this selection a recognizable name, choose ***Rename*** from its context menu and change the name to **Northern Boundary**.

Afterwards, it is important to select (click on) ***Domain*** in the *Spatial Units* panel again. This will make sure that any model properties are plotted on the entire model domain later.

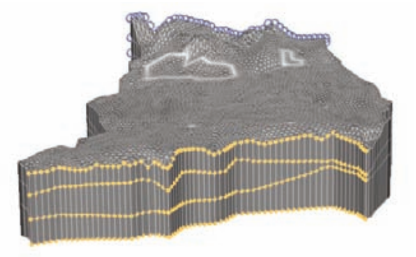
Click on ***Clear selection***.



Southern Boundary

The assignment of boundary conditions along the southern boundary is done in the same way:

- In the *Slice* view, zoom/pan to the southern border.
- *Select nodes along a border*.
- *Copy selection* to all slices.
- Switch to the *3D* view and check that the selection is set correctly.
- Make sure that *Hydraulic-head BC* in the *Data* panel is still active.



- Type **32.1** m in the input box of the *Editor* toolbar and click *Assign*.
- Store the selection and rename it to **Southern Boundary** for later use.
- Select (click on) *Domain* in the *Spatial Units* panel.
- *Clear selection* afterwards.

exercise_fri5.fem

Remaining outer boundaries

At nodes without an explicit boundary condition, FEFLOW automatically applies a no-flow condition. Therefore, no further action is required for the western, eastern, top and bottom model boundaries, which are assumed to be impervious except for groundwater recharge to be added later.

Pumping wells

The wells are to be set in the southern part of the study area based on the map *wells*. They are assumed to be screened throughout the whole depth of the model.

This kind of well, which stretches along a number of layers, is called a multilayer well. Multilayer wells are assigned along vertical element edges (called Join Edges). Nodes along these edge selections are connected automatically by a high-conductive finite element that mimics the borehole.

Demonstration Exercise

Several parameters are necessary to assign a multi-layer well, including the pumping rate, the radius of the well and elevation of the top and bottom end of the screen.

While it is also possible to manually enter these values, it is more convenient to import them from a map.

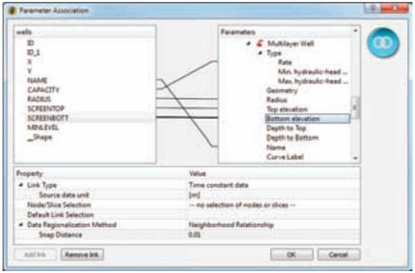
The *wells* map contains attribute data that need to be associated with (linked to) their respective FEFLOW parameter. In order to do this, open the context menu of the map *wells* (in the ***Maps*** panel) with a right click and choose ***Link to Parameter…*** to open the ***Parameter Association*** dialog.

On the left-hand side of the dialog, you see the available data of the map.

The attribute ***CAPACITY*** relates to the abstraction rate of the well, select it with a mouse click.

On the right-hand side, open the ***Boundary Conditions*** > ***Fluid Flow*** > ***Multilayer Well*** branch and click on ***Type*** > ***Rate***.

Click on ***Add Link*** to establish a connection between the values in the map and the Multilayer well or double click on ***Rate*** to set the link.



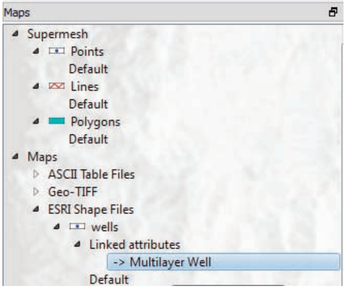
Repeat these steps for the attribute fields

- ***RADIUS*** (link to ***Radius***)
- ***SCREENTOP*** (link to ***Top elevation***)
- ***SCREENBOTT*** (link to ***Bottom Elevation***)
- ***NAME*** (link to ***Name***).

Each well will be created along that edge that is closest to its corresponding data point, but still within a user-defined search radius (snap distance). The ***Snap Distance*** should be small but greater than zero. enter **0.01** meters in the input field. Click the ***OK*** to close the dialog.

The actual assignment is done in a similar way already performed while importing the elevation data.

- In the *Maps* panel, open the branch ***Maps*** > ***ASCII Tables*** > *wells*. Under ***Linked Attributes***, double-click on ***-> Multilayer Well***.



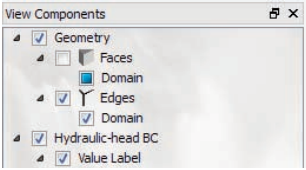
- Click on ***Select All*** in the ***Selection*** toolbar.
- Click on ***Assign*** in the ***Editor*** toolbar. Even though all nodes have been selected, the multilayer wells will be assigned each to the closest node only.

exercise_fri6.fem

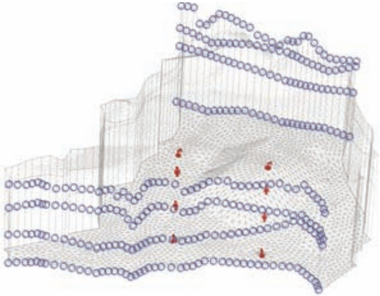
Finally check all boundary conditions you have set.

Clear the selection and go to the 3D view. Make sure that ***Domain*** is selected in the ***Spatial Units*** panel. Double-click on ***Boundary Conditions (BC)*** > ***Fluid flow*** in the ***Data*** panel.

All boundary conditions are shown in the view. Uncheck the checkbox of ***Geometry*** > ***Faces*** in the ***View Components*** panel to see into the domain.



Blue circles are shown on the northern and southern border, and four red symbols for each well.



exercise_fri7.fem

II.5.3 Material properties

Top Aquifer (Layer 1)

In the top aquifer, the hydraulic conductivity, the porosity as well as the groundwater recharge are to be set.

Although groundwater recharge from a mathematical point of view is rather a boundary condition, it is handled as a material property in FEFLOW. In a 3D model, the respective parameter to be set is ***In/Outflow on top/bottom***.

The input procedures for material properties are completely analogous to the ones for the boundary conditions; material properties however are assigned to elements instead of nodes:

- Go to the ***3D*** view.
- Reactivate the checkbox of ***Geometry*** > ***Faces*** in the ***View Components*** panel again.
- Activate ***Material Properties*** > ***Mass Transport*** > ***In/Outflow on top/bottom*** in the ***Data*** panel with a double-click.
- Choose the ***Select Complete Layer/Slice*** tool (in the dropdown selector on the left of the toolbar) in the ***Selection*** toolbar and select all elements in the top layer by clicking on it.
- In the ***Data*** panel, right-click on ***In/Outflow on top/bottom*** and choose ***Set Unit*** > ***mm/a*** from the context menu.
- Input a value of **195** mm/a into the box in the ***Editor*** toolbarand hit <*Enter*>.

The porosity is applied to the same selection by

- Activate ***Drain/Fillable Porosity*** in the ***Data*** panel with a double-click.
- Input a value of **0.1** and hit <*Enter*>.

The hydraulic conductivity will be assigned by interpolating data from field samples.

- Add the map **conduc2d.trp** to the ***Maps*** panel (***Add maps(s)***). This shape file contains point-based conductivity values for the top aquifer in 10^{-4} m/s.
- Associate (***Link to Parameter…***) the attribute column ***Value*** to the FEFLOW parameter ***K_xx***. ***K_yy*** and ***K_zz*** will be calculated from ***K_xx*** and assigned later).
- Choose ***10-4 m/s*** as ***Source data unit***
- As a regionalization method, choose ***Akima***, ***Linear***, with ***Neighbors*** **3** and **0** % ***Over-/Under Shooting***. Activate the checkbox ***Logarithmic***. Close the dialog with the ***OK*** button.
- Double-click on ***Conductivity*** > ***K_xx*** in the ***Data*** panel.

If the top layer is no longer selected, choose the *Select Complete Layer/Slice* tool in the *Selection* toolbar and select it.

- In the *Maps* panel, double-click on *Linked Attributes* > *Value* -> *K_xx*.
- To finally assign the conductivity values, click the *Assign* button in the *Editor* toolbar.

exercise_fri8.fem

Aquitard (Layer 2)

In the second layer, we only need to assign constant values for the conductivity and drain-/fillable porosity.

- *Clear selection* and select the elements in the second layer applying *Select Complete Layer/Slice* again.

A very efficient way to assign multiple model properties is by right-clicking on *Material Properties* > *Fluid Flow* in the *Data* panel and choosing *Assign Multiple...* from the context menu.

In the following dialog,

- Enter **1e-6 m/s** for *K_xx* (including the unit!).
- Enter **0.15** for the *Drain-/fillable porosity*
- Uncheck all other properties.
- Click the *OK* button to finalize the assignment.

Lower aquifer (Layer 3)

Repeat the same steps as for layer 2:

- *Clear selection* and select the elements in the third layer applying *Select Complete Layer/Slice* again.
- Choose *Assign Multiple...* from the context menu of *Material Properties* > *Fluid Flow*.
- Enter **2e-4 m/s** for *K_xx.*
- Enter **0.1** for the *Drain-/fillable porosity.*
- Uncheck all other properties.
- Click the *OK* button to finalize the assignment.

exercise_fri9.fem

Anisotropic hydraulic conductivity

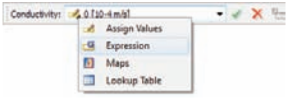
While *K_xx* has been assigned already, *K_yy* and *K_zz* will be derived from *K_xx*.
K_yy is equal to *K_xx*, it will be assigned using a simple copy&paste procedure:

- *Select All* elements in the 3D View.
- Choose *Copy...* from the context menu of *Material Properties* > *Fluid Flow* > *Conductivity* > *K_1m*
- Double-click on *Material Properties* > *Fluid Flow* > *Conductivity* > *K_2m*

- Choose *Paste...* from the context menu of ***Material Properties*** > ***Fluid Flow*** > ***Conductivity*** > ***K_2m***

Besides manual assignment and data import, FEFLOW allows to calculate model properties from mathematical expressions. This will be used for the vertical conductivity ***K_3m***, which is assumed to be 10% of the lateral conductivity ***K_1m***.

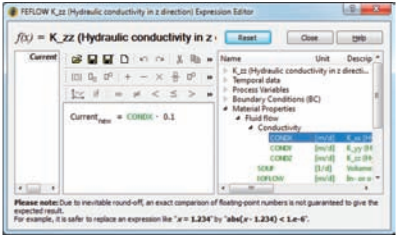
- Double-click on ***Material Properties*** > ***Fluid Flow*** > ***Conductivity*** > ***K_3m***



- Switch to the ***Expression*** input method by clicking on the icon in the input box of the ***Editor*** toolbar or by right-clicking into the box and selecting from the drop-down list. Then double click on ***Current Expression***. The ***Expression Editor*** opens.

The ***Expression Editor*** is a tool to create arbitrary mathematical expressions for various purposes. At the top of the dialog several toolbars provide basic mathematical operations. On the right hand side there is a list of other model parameters that can be used within the formula. All operations and parameters can be added to the formula by a double click, or by directly typing them into the formula using the keyboard.

For this particular operation, select and delete the word "current" on the right hand side of the equal sign. Afterwards, double-click on ***Material Properties*** > ***Fluid Flow*** > ***Conductivity*** > ***CONDX*** in the list on the right to insert it into the formula. Afterwards, click on the multiplication symbol in the toolbar on the top and finally type **0.1** on your keyboard. The resulting expression (see figure) calculates the vertical conductivity as 10% of the horizontal conductivity. Afterwards, press the ***Close*** button.



To finally assign the new values, click the ***Assign*** button in the ***Editor*** toolbar.

Clear selection.

exercise_fri10.fem

Reference Field Data

To be able to compare the computed groundwater levels to measurements, a couple of observation points will be loaded into the model.

Go to the ***Maps*** panel and use the context menu (***Add Map(s)...***) to add **observation_wells.dat** to the list of loaded maps. It is not necessary to visualize the map in the view.

Open the ***Slice*** view again if it has been closed.

Right-click on the map entry in the ***Maps*** panel and choose ***Convert to ...*** > ***Observation Points*** from the context menu. The map file contains information about the location, slice number and measured hydraulic head in the observation well. As default headers are used in this example file, the association of attributes to the properties of the observation points works automatically and no changes need to be done in the upcoming dialog. Click the ***OK*** button to proceed.

The now imported observation wells can be shown by a double-click on ***Observation Points*** in the ***Spatial Units*** panel.

exercise_fri11.fem

II.6 Simulation

The flow part of the flow and transport model is complete. By running the steady-state model, a hydraulic head distribution will be computed that will also act as initial condition for the following transient simulation.

In case that FEFLOW is run in licensed mode, save the model to be able to return to the initial properties later! If running FEFLOW in demo mode, this model cannot be saved as the number of nodes per slice exceeds the allowed maximum of 500. Please use the prepared file **exercise_fri11.fem** in this case.

Starting the simulator

To run the simulation, click ***Start*** in the ***Simulator*** toolbar.

As the model is unconfined, the system is solved iteratively, taking into account that the saturated thickness of unconfined layers depends on the actual solution for hydraulic head. The ***Error Norm History*** chart provides information about the remaining error in each simulation iteration. The simulation stops after eight iterations, the error reaching values below the defined error criterion.

During and after the simulation, all visualization tools in FEFLOW can be used to monitor and post-process the simulation results.

Make sure to select ***Domain*** again in the ***Spatial Units*** panel.

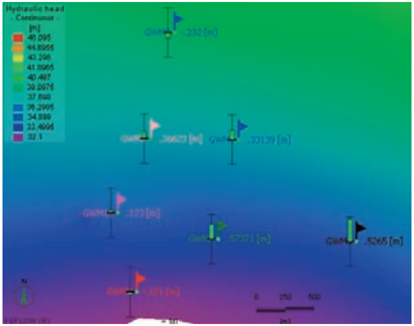
It is recommended to save the file before starting the simulation (if working with a license). During the simulation, the process variables will change and you would lose the initial conditions of the model.

Go to the ***Data*** panel and double-click on ***Process Variables*** > ***Fluid Flow*** > ***Hydraulic head*** for visualization of the resulting hydraulic head distribution.

In addition, go to the ***Spatial Units*** panel and select (single click) ***Model Locations*** > ***Observation Points***. Afterwards, double click ***Process Variables*** > ***Fluid Flow*** > ***Hydraulic head*** in the ***Data*** panel.

(To achieve better visibility in print, the element edges have been removed in the figure).

Make sure to select ***Domain*** in the ***Spatial Units*** panel afterwards.

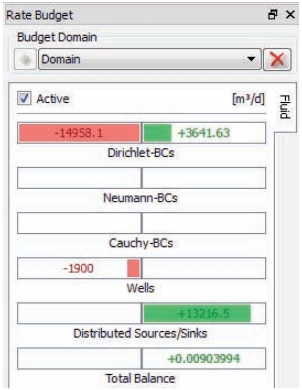


exercise_fri12.fem (To obtain the correct status, run the simulation before proceeding.)

Budget

To check whether the model indeed has reached the steady state, the overall water balance is calculated. The ***Rate-Budget*** panel provides the means for this.

If it has been closed accidentally, open the ***Rate-Budget*** panel via ***View*** > ***Panels*** > ***Rate-Budget Panel***.



Check the ***Active*** checkbox to activate the budget calculation. The budgeting is turned off

On computers with multi-core CPUs or multiple CPUs the simulation result and budget result may be slightly different in each run - even with identical input parameters. This is due to possibly different summation order when using parallelization.

by default as it can cause significant computational effort, especially when being done at each time step during a simulation run.

The budget shows inflows in green, outflows in red for the different boundary condition types and the areal sources and sinks (groundwater recharge). The ***Total Balance*** value - the sum of all water flows - is sufficiently small to accept the solution as steady state.

Streamlines

One way to visualize the flow field is the plotting of streamlines.

Streamlines are calculated by tracking the path of virtual particles that are released at certain starting points (called seeds).

In our case, multiple streamlines are released from around the nodes along the well screens.

First, a selection is created containing all nodes along the well screens.

Go to the ***3D*** view.

In the ***Selection*** toolbar make sure that the ***Select Nodes*** option is set.

In the ***Data*** panel, right-click on ***Boundary Conditions*** > ***Fluid Flow*** > ***Multilayer Well*** and choose ***Convert Parameter to*** > ***3D Nodal Selection*** from the context menu. To save this selection, go to the ***Spatial Units*** panel and right-click to open the context menu.

Choose ***Store Current Selection*** first and then ***Rename*** the saved selection to **Wells**.

Now, uncheck ***Geometry*** > ***Faces*** in the ***View Components*** panel to be able to see inside the model domain.

In the ***Spatial Units*** panel, click on ***Node Selections*** > ***Wells***.

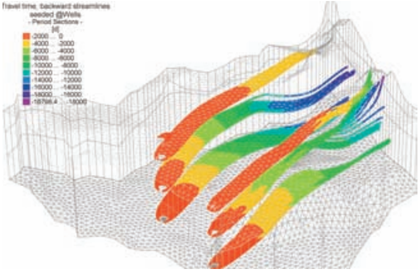
In the ***Data*** panel, double-click on ***Process Variables*** > ***Fluid Flow*** > ***Streamlines*** > ***Backward***.

In the ***View Components*** panel, right-click on ***Travel time, backward streamlines seeded@wells*** and choose ***Properties*** from the context menu.

In the now opened ***Properties*** panel, enter **100** m as the ***Radius***, press *<enter>* and click ***Apply***. Still in the Properties panel, right-click on the color scale on the left and choose ***Presets...*** > ***Rainbow***.

Finally, activate the checkmark ***Travel time, backward streamlines seeded@wells*** > ***Period Sections*** in the ***View Components*** panel.

As a result, the pathlines are shown in the ***3D*** view. The color scale displays discrete intervalls of travel time along these pathways.



From the result it can be seen that the western well is certainly influenced by the sewage fields. The eastern well seems not to be influenced by either contamination source. However, the streamline-based evaluation does not take into account any mixing processes due to dispersion.

For this purpose, a transport model is needed.

Clear selection.

Reactivate the checkbox of ***Geometry*** > ***Faces*** in the ***View Components*** panel before proceeding.

II.7 Flow and Transport Model

To be able to apply changes to the model, click the ***Stop*** button in the ***Simulator*** toolbar.

When the flow model has been run, the process variable ***Hydraulic head*** has changed. After the run, it does not contain the initial conditions any more, but the final results.

In our case these will be used as the initial condition for the transient flow model.

exercise_fri13.fem (To obtain the correct status, run the simulation before proceeding and click on ***Stop***.)

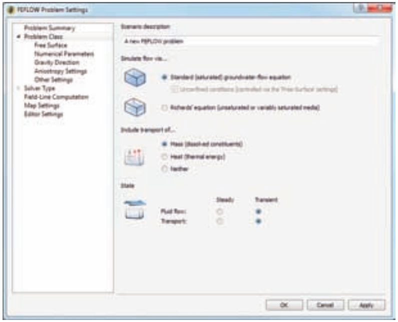
II.7.1 Problem settings

Problem class

From the preliminary streamline analysis based on the results of the flow model it cannot be excluded that the contamination sources are located within the catchment zone of the production wells. To be able to provide quantitative estimations, the model is extended to a flow and mass transport model.

To change the problem class, go to *Edit* > *Problem Settings* and open the *Problem Settings* dialog. In *Problem Class*, select *Mass (Dissolved Constituents)* and choose the *Transient* option for both the *Fluid Flow* and the *Transport* simulation.

Confirm with *Apply*.



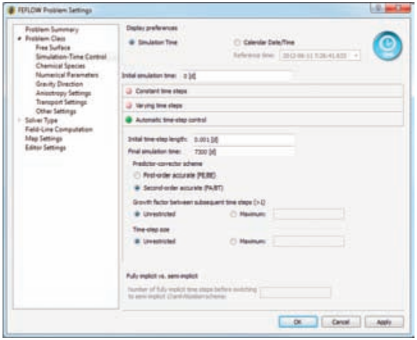
Time stepping

In a transient model temporal discretization has to be defined. The corresponding settings can be found on the *Simulation-Time Control* page.

By default, FEFLOW uses an automatic time-step control scheme. Hereby, an appropriate time-step length is determined internally by monitoring the changes in the primary variables (hydraulic head and concentration).

Enter a value of **7300** days in the *Final Time* input box.

Click on *Apply* and *OK*.



exercise_fri14.fem

II.7.2 Initial conditions

Click on *Domain* in the *Spatial Units* panel.

Switch to the *3D* view. Double-click on *Process Variables* > *Mass transport* > *Mass Concentration* in the *Data* panel. The *3D* view shows the default initial concentration of 0 mg/l, representing fresh water. No changes are necessary for the background concentration.

Contamination sources

The contamination sources are represented by a higher initial concentration in the areas of the sewage fields and the landfill within the first aquifer.

First, a selection of the nodes belonging to the contamination source is needed.

Go to the ***Slice*** view and browse to ***Slice 1***. Double-click on ***Process Variables*** > ***Mass transport*** > ***Mass Concentration*** in the ***Data*** panel. In the ***Maps*** panel, activate (double-click) the map *contamination_sources*. Choose the option ***Select by Map Polygon*** from the dropdown selector of the ***Selection*** toolbar. Make sure the ***Snap*** distance is set to **0** m in the ***Snap-Distance*** toolbar. Click ***Select by All Map Geometries*** in the ***Selection*** toolbar.

The contamination in both areas is found to reach down to the top of the aquitard. Use ***Copy Selection to Slices/Layers*** to copy the selection to slice 2.

The initial concentrations in these areas are to be interpolated from observed data.

Go to the ***Maps*** panel and add the map file **conc_init.shp**.

Associate (***Link to Parameter...***) the attribute column ***CONC*** to the FEFLOW parameter ***Process Variables*** > ***Mass transport*** > ***Mass concentration*** by defining the link. Choose ***Inverse Distance*** as the ***Data Regionalization Method*** and use **4** ***Neighbors*** and ***Exponent*** of **2**.

Click on ***OK***.

Double-click on ***CONC -> Mass concentration*** in the ***Maps*** panel and assign the values by clicking on ***Assign***.

Clear selection.

exercise_fri15.fem

II.7.3 Horizontal Refinement

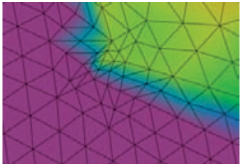
Due to the advective flow term, transport models typical require a finer discretization than flow models. For this reason, the mesh will be horizontally refined around the contamination sources.

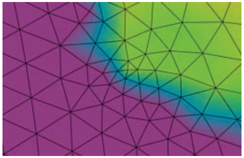
First, the area to be refined is selected as a nodal selection. Increase the ***Snap*** distance to **200** m in the ***Snap-Distance*** toolbar. Choose ***Select by Map Polygon*** in the ***Selection*** toolbar and click on ***Select by All Map Geometries***. This will select all nodes within and up to a distance of 200 m around the contamination sources. Hit the ***Refine*** button in the ***Mesh-Geometry*** toolbar once.

At the transition between the refined and non-refined area, the elements are now quite irregularly shaped (large angles). To improve the mesh quality after manual refinement, the mesh will be smoothed in this area. Make sure that the ***Add to Selection*** option in the ***Selection*** toolbar is active. Define a ***Snap*** distance of **500** m and click ***Select by All Map Geometries*** again in order to create a selection covering the refinement zone and its adjancent area. Press the ***Smooth Mesh*** button to perform the smoothing.

Clear selection.

exercise_fri16.fem





II.7.4 Boundary Conditions

Northern and southern boundaries

Any water entering the domain through the northern or southern boundary is fresh water with a concentration of 0 mg/l. Therefore a fixed concentration of 0 mg/l is assigned as a boundary condition at these locations.

Go to the ***Spatial Units*** panel and open the context menu of the previously stored node selection ***Northern Boundary***. Choose ***Add to current selection***.

Repeat this step with the node selection ***Southern Boundary***.

In the ***3D*** view all nodes along both borders are shown as selected. Click on ***Domain*** in the ***Spatial Units*** panel.

Double-click on ***Boundary Conditions*** > ***Mass Transport*** > ***Mass-Concentration BC*** in the ***Data*** panel. In the ***Editor*** toolbar, ensure that the Assign Values mode is active, input a value of **0** mg/l and click ***Assign***.

Blue circles indicate that first kind boundary conditions are set, similar to the flow boundary conditions.

Constraints

As stated before, water entering the model at the northern or southern border is fresh water. Depending on the hydraulic head distribution, however, at these boundaries also outflow is possible. In this case, a free outflow of contaminated water is to be preferred over applying a fixed concentration.

*Do not forget to select the **Domain** entry again in the **Spatial Units** panel before continue (otherwise no model properties can be plotted in the active view)!*

This requires a dynamic change of the mass transport boundary condition depending on the flow direction, which can be implemented by applying a constraint.

A constraint in our case is used to limit the mass flow at a 1st kind boundary condition (fixed concentration) to a minimum or maximum value. For this exercise, the constraint is set to limit the mass flux to a minimum value of 0 g/d, applying the concentration boundary condition only for inflowing water.

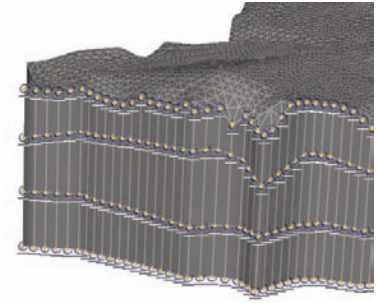
Algebraic signs are handled differently for Constraint and Boundary Conditions: For Constraint Conditions inflows are positive (+), outflows are negative (-). For Boundary Conditions Inflows are negative (-), outflows are positive (+).

The constraints are technically applied in the same way as boundary conditions. However, for the sake of clarity, the constraint conditions are not shown by default as properties in the ***Data*** panel and have to be added first:

In the ***Data*** panel, open the context menu of ***Boundary Conditions*** > ***Mass transport*** > ***Mass-Concentration BC*** and choose ***Add Constraint*** > ***Min. mass-flow constraint***.

Expand the tree view and activate (double-click) ***Min. mass-flow constraint*** and assign **0** g/d to all the selected nodes with *<Enter>*. The minimum constraint is indicated by a bar below the associated boundary condition symbol.

Clear selection.



exercise_fri17.fem

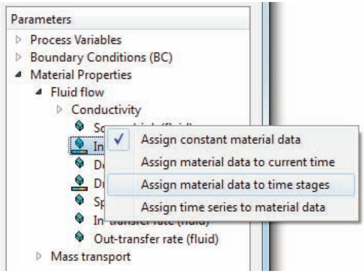
II.7.5 Material properties

As the annual rainfall data shows a significant variability during the simulated period, the groundwater recharge is assumed to be time-varying in the model. The file **recharge_annual.shp** contains the spatial distribution of the approximated

recharge for annual periods each in a separate attribute field.

Go to the ***Maps*** panel and load the map **recharge_annual.shp** (right-click and choose ***Add Maps...***) and choose ***Link to Parameter...*** from its context menu.

In the ***Parameter Association*** dialog, browse to ***Material Properties*** > ***Fluid Flow*** > ***In/Outflow on top/bottom*** on the right and open the context menu with a right-click. Choose the option ***Assign Material Data to Time Stages***. The upcoming dialog lets you define the time stages for which time-varying recharge data shall be assigned (for time steps between these time stages, the recharge is temporally interpolated during the simulation).

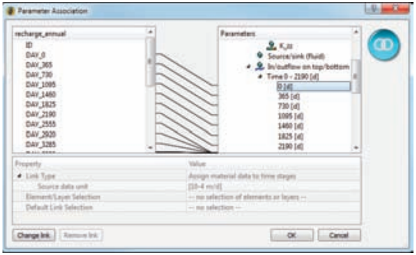


The list has to contain the same values as given in the attribute fields of the map (0;365;730;...). Instead of populating the list manually, the ***Import...*** button lets you get those time stages automatically from the pre-defined file **recharge_time_stages.pow**. Choose this file in the file selector dialog and confirm with the ***Open*** button.

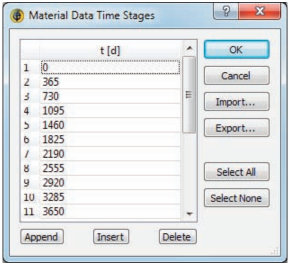
This will fill the list with annual time stage intervals up until 7300 days. Click ***OK*** to close the dialog.

The links between the attribute fields and the time-stages is done in the same way as with constant model properties. However the work can be accelerated by creating a multiple selection of all attribute fields (0 .. 7665) before creating a link with the 0 [d] time stage. In this case, FEFLOW will automatically create links between all subsequent attribute fields and time stages (see figure).

Click the ***OK*** button to close the dialog.



Go to the ***Slice*** view and browse to Slice 1.



In the *Maps* panel , double-click on *year_rec* > *Linked Attributes* -> *In/Outflow on top/bottom* and *Select All* elements. Finally, set the *Snap* distance to **0** m and assign the values by clicking on *Assign*.

The values for the time stages have now been imported. When looking at *In/outflow on Top/Bottom* in the *Data* panel, notice that a tilde symbol marks the material property as time varying.

Clear Selection.

Visualize the different recharge values for time stages by right-clicking on *In/Outflow on Top/Bottom* in the *View Components* panel and choosing one of the entries in *Material Time* in the context menu.

exercise_fri18.fem

To simplify the data input of the remaining parameters, the material parameters effective for mass transport processes (porosity as well as longitudinal and transversal dispersivity) are assumed to be homogeneous throughout the model.

Go to the *3D* view and activate (double click) *Material Properties* > *Mass transport* > *Porosity* in the *Data* panel.

Right-click on *Material Properties* > *Mass Transport* in the *Data* panel and choose *Assign Multiple...* from the context menu.
Afterwards,

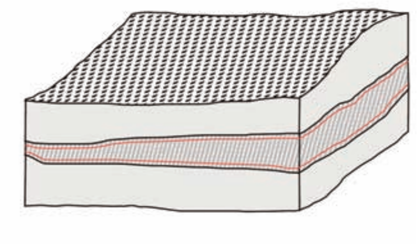
- enter **0.2** for *Porosity*
- enter **70** m for *Longitudinal dispersivity (mass)* and
- enter **7** m for *Transverse dispersivity (mass)*
- Deactivate all other entries and click the *OK* button to finalize the assignment

exercise_fri19.fem

II.7.6 Vertical resolution

To ensure a correct representation of low flow velocities in the aquitard as a basis for transport simulation, this model layer has to be further subdivided.

The best choice to minimize errors due to the nodal nature of the calculated velocity field is to apply thin layers on top and bottom of the aquitard.



The two additional slices are placed within the aquitard with a distance of 10 cm to the aquitard top and bottom.

Go to *Edit* > ***3D Layer Configuration***.

Type a value of **0.1** m in the ***Distance*** input box and - for inserting the first additional slice - choose the option ***lower slice***.

Increase the ***Number of layers*** in the input box in the upper left corner of the dialog by one (to **4**). Press *<Enter>*. The ***Slice Selection*** dialog is opened.

On the right side, a list of the existing slices is given. The new slices (from the list on the left) are to be included here via drag and drop.

Drag the ***New slice 1*** between ***Slice 2*** and ***Slice 3*** on the right. Click on ***OK***.

The new slice is inserted into the model with the slice number 3. Set its status to ***Fixed***, checking the checkbox.

The next slice is to be inserted 0.1 m below the second slice. Change the option to ***upper slice***.

Increase the ***Number of layers*** to **5** and hit *<Enter>*.

In the upcoming ***Slice Selection*** dialog, drag the ***New Slice 1*** again between ***Slice 2*** and ***Slice 3*** and click on ***OK***.

The aquitard has now been divided into three layers. To ensure that the data are transferred correctly from the old to the new slices and layers, have a look at the ***Data Flow*** lists on the right of the ***3D Layer Configurator***.

There are two lists that provide control over the data flow between the previous and the new slices and layers. The upper control called ***Data flow for slices*** describes the data flow of the process variables, boundary conditions, and nodal selections from the old slices to the new ones. The old slices

are shown as number buttons in the left column, the new ones in the right column. The data flow is symbolized by lines connecting the old with the new slices. The lower list, ***Data flow for layers*** describes the data flow for all material data and for elemental selections.

The nodal parameters of the old slice 2 are inherited to the new slices 2 and 3, while the information from the old slice 3 is to be inherited by slices 4 and 5.

FEFLOW suggests to transfer the model properties from old slice ***2*** to slices ***2*** – ***4***. In order to change this, double click on the number box showing ***5*** in the new column. Change this entry to 4 – 5.

As a result, the link from old slice ***2*** points to new slices ***2-3*** and from old slice ***3*** to new slices ***4-5***.

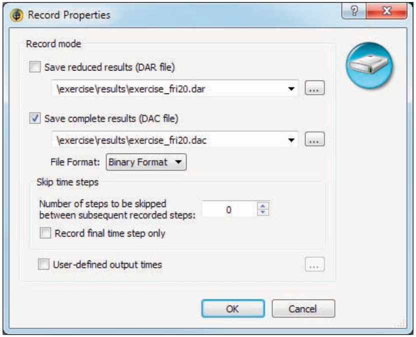
The data flow in the lower list for the material properties describes the same data characteristics from the old bottom layer (lower aquifer) to the new layers 2, 3 and 4. No changes are necessary.

Click on ***OK*** to exit the ***3D Layer Configurator*** and to apply the changes to the model.

exercise_fri20.fem

II.7.7 Simulation Run

If working with a licensed version of FEFLOW, the results can be saved to a file during the simulation run. Click on ***Record*** in the ***Simulator*** toolbar. Activate ***Save complete results (DAC file)***. By default, the results file (*.dac) is saved with the same name as the current model in a subdirectory ***results***. Confirm by clicking ***OK***.

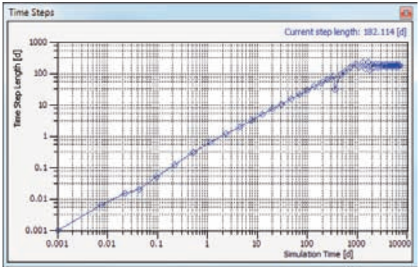


If running FEFLOW in demo mode, it is not possible to save the results file, and only the prepared file (**exercise_fri20.fem**) can be run.

To run the model, click ***Start*** in the ***Simulator*** toolbar. The simulation takes approximately 5 minutes on a system with an Intel i7 processor.

The current simulation time is displayed in the dropdown box of the ***Simulator*** toolbar.

In the ***Time-Steps History*** (which can be opened via ***View*** > ***Charts*** if not already shown) the actual time step length versus the total simulation time is plotted. The mostly steady conditions lead to a steadily increasing time step length, with a reduced time-step whenever a change of the groundwater recharge occurs.



The simulation stops after 7,300 days (the final time that has been set in the ***Problem Settings*** dialog before).

Click ***Stop*** to leave the simulator.

II.7.8 Postprocessing

Load the recorded results file.

exercise_fri20.dac

Demonstration Exercise

Slice View

Go to the ***Slice*** View and double click on ***Process Variables*** > ***Mass Transport*** > ***Mass Concentration*** in the ***Data*** panel.

Clear selection if necessary.

The spatial distribution of the Nitrate concentration at the final simulation time is shown.

Cross Sections

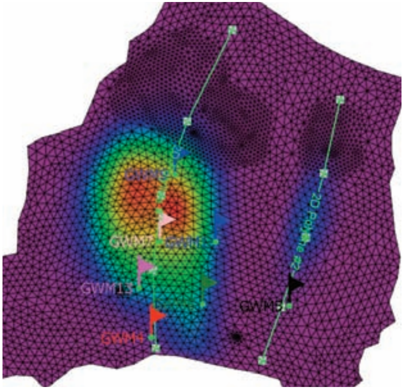
Process variables and material properties can be visualized in cross-section views.

In our case, we are interested in the vertical distribution of the contamination along the plumes.

The cross section is based on a 2D Surface Line. Switch to the ***Slice*** view. Choose ***Draw a Surface (2D) Line***.

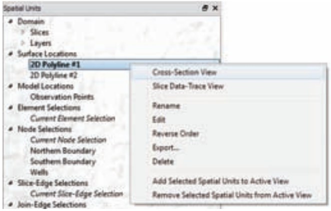
Click on the sewage fields (the contamination source in the north-west) to define the starting point of the line here. The line is extended by adding points with a single mouse click. Follow the flow path to the western well and further to the lake Müggelsee. Finish the line with a double click.

Repeat these steps for the waste dump (the eastern contamination source). The final result should look similar to this:



In the ***Spatial Units*** panel, two new entries ***Surface Locations*** > ***2D Polyline #1*** / ***#2*** have been added.

Open the context menu of ***2D Polyline #1*** and choose ***Cross-Section View***.

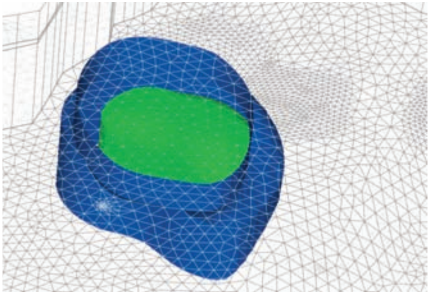


A cross-section view showing the depth-related concentration along the cross-section is opened.

In the ***Navigation*** panel, go to the ***Projection*** tab and push up the lever to exaggerate the z-axis.

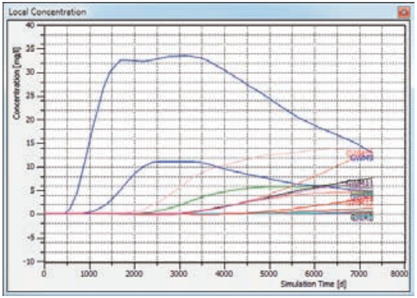
Isosurfaces

Go to the ***3D*** view. Make sure that ***Domain*** is selected in the ***Spatial Units*** panel. In the ***Data*** panel, double-click on ***Mass concentration*** to show this parameter in the view. In the ***View Components*** panel, uncheck ***Faces*** and ***Mass concentration*** > ***Continuous***. Check ***Mass concentration*** > ***Isosurfaces*** > ***Domain***. Instead, one isosurface is shown.
To edit the isosurface visualization properties, double-click on ***Isosurfaces.*** The ***Properties*** panel comes to front. Switch to the ***Custom*** mode and click on ***Edit.*** Specify two values, **10** mg/l and **20** mg/l. Close the dialog by clicking ***OK,*** and click ***Apply*** in the ***Properties*** panel. The isosurface visualization is changed to reflect the newly set concentrations.



Breakthrough Curves

Open from the menu ***View*** > ***Charts*** > ***Local Concentration History***. The diagram shown contains the concentrations calculated at the different observation points during the simulation time.



More Information

This completes the demonstration exercise, that gives an introductive overview of the basic functionality and workflows of FEFLOW.

Additional tutorials, application examples and more detailed descriptions of the program features are available in the User Manual.

For more information, including extensions, tutorial videos, user forum and more, please visit our website

www.feflow.com